

Gas Generator Induced Flow and its Effect on Fire Flame Extinction

Pierre Joulain
ENSMA Universite de Poitiers

Jose L. Torero
University of Maryland
College Park, MD 20742-3031



United States Department of Commerce
Technology Administration
National Institute of Standards and Technology

Gas Generator Induced Flow and its Effect on Fire Flame Extinction

Prepared for

U.S. Department of Commerce
National Institute of Standards and Technology
Gaithersburg, MD 20899

By

Pierre Joulain
ENSMA Universite de Poitiers

Jose L. Torero
University of Maryland
College Park, MD 20742-3031

December 1997
Issued April 1998



Notice

This report was prepared for the Building and Fire Research Laboratory of the National Institute of Standards and Technology under grant number 70NANB7H0005. The statement and conclusions contained in this report are those of the authors and do not necessarily reflect the views of the National Institute of Standards and Technology or the Building and Fire Research Laboratory.

Gas Generator Induced Flow and its Effect on Fire Flame Extinction

Pierre Joulain

Visiting Researcher-BFRL-NIST and
Visiting Professor Department of Fire Protection Engineering, UMCP
Laboratoire de Combustion et de Detonique-UPR 9028 au CNRS
ENSMA-Universite de Poitiers
Teleport 2 – BP109, 86960 Futuroscope Cedex - France

and

Jose L. Torero

Department of Fire Protection Engineering
University of Maryland
College Park, MD20742-3031

Final Report

Contract No. 70NANB7H0005

February 1998

Prepared for:

U.S. Department of Commerce
National Institute of Standards and Technology
Building and Fire Research Laboratory
Gaithersburg, MD20899-0001

ABSTRACT

A limiting factor to the development of new suppression technology is the lack of appropriate screening methods. Among the new technologies that are intended to replace Halon1301 are Solid Propellant Gas Generators, (SPGG's), or flame suppressing gas generators. SPGG's are a spin-off from airbag technology and have demonstrated their ability to suppress certain types of fire, particularly aircraft engine nacelle and dry bay fires. To date there is no adequate screening method for SPGG's. The flow originating from burning a solid propellant reduces the Damköhler number by decreasing the residence time (high velocity flow of products) and increasing the chemical time (by directly altering the reactant concentrations, oxygen displacement effect). If the Damköhler number decreases below a critical value sudden extinction of the flame occurs. A facility that will serve to assess the performance of SPGG's has to be able to evaluate the combined effects of the gas discharge. Several alternatives have been proposed for an adequate flame and enclosure that will represent a "worst case" scenario for extinction and subsequent re-ignition. Among these alternatives is the recirculation zone induced by either a bluff body, a baffle, a backward facing step or a trench inside a wind tunnel. These configurations provide a controlled increase in the residence time, thus an increase in the Damköhler number. This report describes a preliminary evaluation of these configurations by means of a literature search and some preliminary computations using Large Eddy Simulation (LES) code developed at NIST. The literature review showed that the proposed configuration is adequate for the present application since it creates a recirculation zone with enhanced mixing, entrainment from the main stream and product evacuation. The flow parameters can be adjusted to provide a stable recirculation zone in a broad range of main stream velocities. Limitations to this approach together with a series of design criteria are also proposed. Preliminary calculations with the LES code showed qualitative agreement with reported experimental data while the literature showed that $k-\epsilon$ codes seem inappropriate to model the recirculation zone. Different diagnostic techniques used for similar experiments are evaluated as candidates for characterization of the experimental facility, emphasis is given to velocimetry. Finally, an intermediate scale experimental facility at the LCD-University of Poitiers, France, is presented. This fixture has the potential to serve as an intermediate scale validation of the proposed screening procedure.

ACKNOWLEDGEMENTS

This work was funded by NIST. The authors will like to acknowledge the support of Dr. William Grosshandler and the extensive help and technical information provided by Dr. Kevin McGrattan when using the L.E.S. code. The computational work was conducted by John Krawiec.

TABLE OF CONTENTS

ABSTRACT	iii
ACKNOWLEDGEMENTS	iv
TABLE OF CONTENTS	v
LIST OF FIGURES	vi
1. Background	1
2. Introduction	4
3. Solid Propellant Gas Generators (SPGG's)	7
4. Literature Review	12
a. Structure of the Flow-Field Behind a Step	12
b. Heat Transfer Characteristics Behind a Step	18
c. Ignition of a Flammable Mixture Over a Hot Plate	20
5. Diagnostic Techniques	21
a. Velocity Measurements	21
b. Time Dependent Density Measurements	24
c. Heat Transfer to the Walls	24
6. Characteristics of the Poitiers' Intermediate Scale Fixture	25
7. Numerical Methodology	28
8. Conclusions	30
9. References	33

LIST OF FIGURES

Figure		Page
1	Normalized size of the recirculation zone as a function of the Reynolds number	16
2	Experimental apparatus at the University of Poitiers	25
3	(a) Presence probability of the flame in the transversal plane. (b) Mean flame contour.	27

1. Background

When a fire breaks out there is mainly one immediate concern: how to prevent the fire from spreading without casualties and with a minimum property loss. For many years, at least until 1987 and the signature of the Montreal Protocol, other issues, such as the interaction of the suppression process with the local and global environment were a matter of little concern. A good example of highly effective suppression agents affected by the Montreal Protocol is Halon. Recent environmental problems associated with Halons have been well publicized and an amendment to the Protocol required commercial Halon production to cease at the beginning of 1994. However, Halons have been the agents of choice for numerous fire protection applications mainly because of their great ability to inhibit flames at low concentrations but also because of other positive characteristics (high liquid density, high stability, low boiling point, toxicity, cost, etc....). Consequently large users of Halon have been forced to search for suitable alternatives.

Following the Montreal Protocol there was a renewed interest in defining new fire suppression technology and as a result an early review of the technology available to replace Halon was presented in 1993 [1]. DiNenno [2] and Tapscott [3] focused their analysis on the search for alternative chemical agents. Grosshandler and al. [4,5] discussed research conducted for aircraft applications. New concepts involving chemical agents, water-based options, chemically generated gases and aerosols, and physically-acting suppression were presented. A summary of the results obtained from extensive laboratory-scale experiments was also included.

It has been experimentally demonstrated that solid propellant gas generators (SPGG's) can be successfully used in a manner similar to a streaming agent to suppress fires while remaining environmentally friendly. The combustion products emerging from gas generators are considered to have no ozone depletion and global warming potential and its use leads to enormous practical advantages such as small size and long storage life. Efforts to make the use of this technology possible have significantly increased in the last few years with significant involvement of both private and public sectors. Notable among these efforts are those of the National Institute of Standards and Technology (Building and Fire Research Laboratories), the Navy, Airforce, aircraft

manufacturers and military contractors. Extensive experimental work has been done by the Naval Air Warfare Center Weapons Division in China Lake. In cooperation with several aircraft manufacturers, experiments were conducted on real scale facilities (High Velocity Airflow System (HIVAS), F/A-18 and V-22 wing dry bay simulators, F/A-18 engine nacelle simulator) leading to the development of gas generators specifically designed for fire suppression (PFE, FSGG). Similar tests have been conducted by the Air Force on CFM-56 engines (found on the KC135-R). Details of the status of these research programs and other research efforts have been summarized in the proceedings of the NIST-1995 Workshop on Solid Propellant Gas Generators [6].

Grosshandler et al. [7] pointed that one of the main limitations to the development of new suppression technology was the lack of appropriate screening methods. They examine two new concepts for testing liquid aerosol and solid propellant gas generator (SPGG's) fire suppression technologies. The first concept is for a bench-scale suppression screen suitable to compare the ability of dispersed fluids with differing chemical and physical properties to extinguish a laboratory flame; the second is for a facility to test SPGG-based agents and release mechanisms. The present review lies in the framework of the development of this second concept.

Definition of a facility that will serve to test SPGG-based agents and release mechanisms requires the definition of an adequate flame that will serve as a standard and characterization of its interaction with the flow induced by a gas generator. The existing knowledge on the characteristics of the flow field in the fire enclosure and the flow induced by a gas generator is so far limited to the needs of the current applications (cooling air, airbag inflators). In general, the available literature is of global nature and consists mainly on average temperature, velocity, pressure and product concentration measurements. This information is important as it relates to the nature of the resulting atmosphere but provide little insight on the mechanism that lead to flame extinction.

Since extinction is most likely to occur during the transient stages and re-ignition might be possible, before steady conditions are attained. To quantify the potential for re-ignition it is necessary to characterize the flow field in the facility and its transient behavior after discharge of the SPGG.

Several alternatives have been proposed for an adequate flame and enclosure that will represent a “worst case” scenario for extinction and subsequent re-ignition. It was determined by Hamins [8] that a controlled increase in the residence time is obtained inside the recirculation zone induced by either a bluff body, a baffle, a backward facing step or a trench inside a wind tunnel. Preliminary computations were conducted by means of a Large Eddy Simulation (LES) CFD code [9,10] with promising results. The base geometry of a wind tunnel with an induced recirculation zone will be explored in detail throughout this report and the possibility of using the LES code in the design process will be assessed.

This work is part of a multi-step study including literature survey, experimental, theoretical and numerical development of the various aspects of the problem: fluid mechanic, combustion and their interactions. This report summarizes a literature review and evaluation of a numerical technique conducted with the objective of providing some background for the design of a well characterized test fixture for screening the fire suppression effectiveness of agents. The facility should be designed with inherent flexibility, allowing variation of key geometric parameters that impact the characteristic residence time of the recirculation zone (the Damkhöler number). Thus, the literature survey focuses primarily on the work related to the description of the structure of the flow field behind a backward-facing step or a baffle. Furthermore, a medium scale fixture used at the CNRS Poitiers to study the interaction between a cross-flow and a pool fire is presented and its ability to serve as an intermediate test facility for SPGG’s is underlined.

2. Introduction

Gas generators discharge the cooled down products of a solid propellant combustion reaction into ambience. A gas generator typically consists of a solid propellant tablet that will, upon ignition, rapidly react to generate gas phase combustion products and particulate, an igniter to initiate the combustion of the propellant, a filter system and an exhaust mechanism. Particulate are trapped by a filter that also serves to reduce the temperature of the gas-phase combustion products. This element has been commonly used as a mechanism to inflate air-bags. The principal gas-phase product of the combustion process is nitrogen, therefore, a logical extension to this technology is fire suppression [6].

If a flame is subject to a fast flow of nitrogen extinction might occur. The advantages of generating this fast flow by means of a solid propellant are many: it is compact, it has a very long storage and service life, it can be used in areas of difficult access, the response time can be extremely fast and this type of system is considered to have no ozone depletion or global warming potential. One of the main issues that has slowed the development of gas generators as a fire suppression technique is the lack of adequate test protocol to assess their performance. Estimation of the reliability and efficiency of this technique depends on a better understanding of the high speed flow and the fundamental chemical and thermal processes involved in extinction and re-ignition of a fire.

The process of extinction can be described by means of the Damköhler number ($Da = (\text{Residence Time})/(\text{Chemical Time})$), by either reducing the residence time or increasing the chemical time a critical Damköhler number for extinction can be attained. The flow originating from burning a solid propellant can reduce the residence time (high velocity flow of products) and increase the chemical time (by directly altering the reactant concentrations, oxygen displacement effect) resulting in sudden extinction of the flame. Since the changes induced by the burning solid propellants are very strong and sudden, the extinction process is expected to be extremely fast. This suppression technique has

potential applications as a substitute to Halon 1301 in engine nacelles, dry bays and army vehicles as well as localized and difficult to access fires.

Extinction mechanisms have been a subject of numerous studies and excellent reviews can be found in the literature. A good summary of the existing knowledge is provided by Williams [11]. The process of extinction can be described by means of the Damköhler number ($Da = (\text{Residence Time})/(\text{Chemical Time})$), by either reducing the residence time or increasing the chemical time a critical Damköhler number for extinction can be attained. The flow originating from burning a solid propellant can reduce the residence time (high velocity flow of products) and increase the chemical time (by directly altering the reactant concentrations, oxygen displacement effect or by decreasing the temperature of the pyrolysis and reaction zones) resulting in sudden extinction of the flame. Furthermore, the interaction between the flame and the flow, coming from the gas generator, can affect the turbulent structure of the system. Local stretch rates can be altered by the nitrogen rich flow resulting in local extinction zones. Depending on the geometry of the system, the type of flame, the characteristics of the gas generator and the distance between the gas generator and the flame the effects of each independent mechanism can be very different.

An issue of great relevance to this specific problem is the relative importance in the extinction process of radical depletion and temperature decrease. The flow induced by the gas generator will affect the chemistry of the reaction as well as decrease the temperature of the flame and fuel. Although the imposition of a non-oxidizing flow and the enhanced heat losses induced by the gas generator tend to, both, favor the extinction process, their relative importance will have a strong effect on the range of scenarios where a gas generator can be used as a fire suppression mechanism. Geometrical considerations will be different for both extinction mechanisms. Significant work on the interaction between flames and high speed flow has been reported in the past and have been reviewed by Williams [11] and Blazowski [12] but all the information pertains to scenarios that significantly differ from that of fire flame extinction.

When considering the sudden onset of an enhanced velocity field past a given object it can be observed that the transient conditions under which the object will continue to burn (or extinguish) differ little from those under which it will continue to burn (or extinguish) in a steady field of equal intensity and distribution. If the flame can not persist after the sudden onset of the velocity it is likely that it will not re-ignite once steady conditions are attained. Cooling down of the fuel will then follow [12]. In the case of solid propellant induced flows the extent and characteristics of the transient and steady state conditions depend on the geometrical configuration. The relative position of the fire with respect to the generator will significantly affect the flow structure and development. If the fire is close to the generator and the characteristic volume of the geometrical boundaries are very large compared to that of the fire and the gas flow induced by the generator, then both the transient and steady state periods will be very short and the flow field will regain its initial characteristics very fast. If the characteristic volume of the flow created by the gas generator is comparable to the volume of the room then a short transient period will be followed by a long steady period (flooding), in this case the flow field will not be expected to regain its initial characteristics. Obstructions in the path between the generator and the fire will significantly affect the velocity and distribution of the flow reaching the fire and will also affect the extent and nature of the transient process.

If flooding occurs it is expected that the critical Damköhler number will be attained permanently and cooling down of the fuel, which is a very slow process (when compared to the chemical and residence times of the flame), will follow. If both transient and steady periods are very short, first scenario depicted above, the flow conditions will provide a Damköhler number that will allow re-ignition if enough gaseous fuel can still be produced when the low velocities and high oxygen concentrations are re-instated [13,14]. Fuel pyrolysis will depend on the surface temperature and for many materials, when burning, the surface temperature is much higher than the vaporization temperature (i.e. charring materials), therefore, fuel pyrolysis will occur even after extinction of the flame. Comparison between the characteristic time of the flow induced by the gas

generator and the characteristic cooling time becomes relevant and re-ignition becomes a significant issue when evaluating the performance of gas generators as a suppression mechanisms.

A simple way to increase the residence time is by creating a recirculation zone, this will significantly reduced the strain on the flame and thus provide a “worst case scenario” to test gas generators. The different issues that need to be addressed when designing such a facility will be covered throughout a review of relevant studies. These studies include work on the flow structure behind a backward facing step, a baffle and a trench. The review is complemented by some background information on gas generators and preliminary CFD results using the LES [9,10] code and validated against experimental data obtained from the literature survey.

3. Solid Propellant Gas Generators (SPGG's)

Solid propellant gas generators, (SPGG's), or flame suppressing gas generators, a spin-off from airbag technologies, have demonstrated their ability to suppress certain types of fire, particularly aircraft engine nacelle and dry bay fires [15]. As stated by Yang and Grosshandler [6] the main areas where research will contribute to the efficient implementation of this technology are:

- Identification of certification procedures for gas generators in fire suppression applications,
- Determination of critical parameters for evaluating fire suppression applications.
- Development of a standard methodology to facilitate testing of gas generators.
- Identification of possible applications other than protection of engine nacelles and dry bays,
- Identification of a new generation of propellants.

According to Yang and Grosshandler [6], a SPGG is essentially an airbag inflator without a bag. That is, the gas generated is discharged directly into ambience rather into a bag. A typical SPGG consists of:

- a solid propellant tablets which will, upon ignition, rapidly react to generate gas-phase combustion products and particulate
- an igniter to initiate the combustion of the propellant
- a filter system to prevent or minimize the release of particulate from the combustion reactions into the ambience
- a heat transfer system to cool the high temperature combustion gas before being discharged into the ambience
- an exhaust mechanism to disperse the gas efficiently.

There are basically two types of airbag inflator systems: the conventional and the pre-pressurized or gas-assisted. In a conventional system, the gas that is used to inflate the bag depends entirely on the combustion gas generated by the solid propellant. However, in a pre-pressurized or gas-assisted system, the high temperature gas as a result of the combustion of the propellant is first mixed with a pre-pressurized inert gas at ambient temperature before being discharge into a bag. Similarly, one can also conveniently classify solid propellant gas generators into two categories, depending upon their functions: conventional and hybrid. When a gas generator is used alone for fire suppression, it is termed “conventional”. When it is used together with other liquid or powdered fire suppressing agents, it is termed “hybrid”. In a hybrid system, the gas generator normally is used as a means to provide sufficient pressurization so that the expulsion of liquid or powdered agent from a storage vessel can be facilitated. A different alternative or hybrid inflator is composed of pressurized argon that is heated by a small charge to produce a sufficient volume of gas for bag inflation [16]

The attractiveness of using SPGG’s in fire suppression applications lies in the fact that the system, when used alone, is considered to have no ozone depletion and global warming potential, and is physically very compact. Being a derivative from the airbag

inflator technology, there are numerous research publications available. Another advantage is that since gases are generated via solid propellant reactions, the system can, in principle, be tailored to function over a period of few milliseconds to few seconds by manipulating the parameters that control the combustion mechanisms. In addition, the gas generators have very extended storage and service life. However the toxicity of some of the by-products cannot be ignored.

Sodium azide is the current principal chemical used in solid propellants for gas generators. Because of its potential health hazards airbag manufacturers have focused recent research on the “non-azide based” propellants. The development of any new propellant should include the improvement of: propellant thermochemistry and stoichiometry, propellant ignitability and burning rates under various conditions, toxicity of combustion products, stability of propellant during storage and transport, and propellant thermal properties. In addition issues that need to be addressed are the grain size, shape of the propellant and how the propellant is packed in the gas generator.

Most relevant information related to SPGG’s is present in the airbag inflators and solid propellant literature. Global characteristics of the performance of automotive airbags address issues specific to this application [16-19] but might be relevant to fire suppression. One of these issues is the rate at which the combustion products are released. Generally this rate is controlled by cutting the sodium azide propellant into pellets. The size of the pellets determines the surface area, which subsequently determines the rate of gas generation. Typical values for gas generation rates result in airbags being inflated in 20 to 40 milliseconds [19]. Also related to the gas generation is the nature of the combustion process. Berger and Butler [20] examine the performance of gas-generating propellants by comparing the theoretical combustion behavior of three condensed-phase propellants commonly used in the airbag industry. They considered an azide and a non-azide propellant, and a double-base propellant. They investigated various thermophysical properties including the flame temperature and chemical composition of the product gases, the number of gaseous moles produced per mass of propellant consumed, the condensed-phase or slag production of each propellant, and the toxicity of the combustion products. These investigations were carried out by means of

the widely used test tank. This work concludes that the thermophysical properties of the propellant have a significant effect in the performance of the SPGG.

Average experimental measurements of the output of a gas generator have always been of questionable validity, mainly due to the unsteady nature of the process [20]. Many attempts have been made to provide local and transient measurements, among the different methods proposed, Wang [21] developed a new semi-analytical procedure, called the dual-pressure method, for computing the output from a pyrotechnic inflator. This method appeared better than the more traditional average temperature method, since it predicts the transient gas temperature in inflators. Although the transient evolution of the pressure can be tracked, the results remain of qualitative nature.

In an attempt to cover the lack of quantitative experimental work modeling has been extensively used to evaluate the performance of SPGG's. Motevalli and Bedewi [22] presented a general overview of the modeling effort in the area of airbag deployment. This review describes the main possibilities of existing codes and the required developments necessary to improve them. This paper points that numerical codes use uniform gas thermodynamic properties and empirical data to calculate gas generation and calculation of an airbag internal pressure. To obtain an adequate characterization of the effluent it is necessary to have the ability to conduct a time-dependent calculation of the gas flow into and out of the airbag that includes pressure interactions. In addition, gas jet effects need to be incorporated which forces the solution of the full Navier-Stokes equations. Furthermore, it was recommended that a sensibility analysis of the current models should be conducted.

Work has been conducted in some of the areas considered of importance by Motevalli and Bedewi [22] but the results are still far from being complete. Lupker and Bruijs [23] and Groenenboom et al. [24] present modeling results that concentrate on the jet issuing from the gas generator and its impact during the airbag inflation. A physical model for a conventional gas generator and a gas assisted pyrotechnic inflator was developed and validated by Krier and Butler [25] and Butler et al. [26].

Butler et al. [26] developed a mathematical model to simulate the transient, thermochemical events associated with ignition and combustion of pyrotechnic

automotive airbag inflator. The conservation equations for mass and energy were derived for the interior combustion chamber, filter/cooling screens, exterior plenum, and discharge tank. After a brief description of the model development and of the physical assumptions made in the analysis, two series of test calculations were presented. The first series of calculations is considered as the baseline case of a conventional pyrotechnic inflator system that is characteristic of a standard discharge tank validation experiment. Transient pressure and temperature profiles generated by the airbag inflator model are presented along with properties at the exit nozzles. The parametric study demonstrates the usefulness of airbag inflator simulation in assessing the sensitivity of airbag pressure curves to various design parameters such as propellant and hardware properties and hardware dimensions. The second series of calculations illustrates the influence of pre-pressurized inert gas on the performance of a pre-pressurized pyrotechnic inflator system. Performance of the inflators is given in terms of pressure-time and temperature-time profiles in the inflator and discharge tank as well as pressure-time integrals at specified time after ignition. The pre-pressurized pyrotechnic inflator shows certain advantages over conventional pyrotechnic units, including significantly lower requirements for solid propellant mass, lower operating temperature, more uniform performance at hot and cold ambient conditions, and higher thermal efficiency. The chemical composition of the inert gas, pre-pressurized system, is also shown to influence the working process of inflator. Validation of these results was made with restricted experimental data. Further development and experimental validation of this model could result in a very useful tool for the design and the development of new inflators, new pyrotechnic compositions, ignitors and filters and also of the propellant grain structure.

In a recent paper Schmitt and al. [27] extend the previous numerical work to study the transient operation of a non-azide propellant pre-pressurized airbag inflator. Predicted performance of the inflator is presented in terms of pressure, temperature and mass flow rate profiles in the inflator and discharge tank which is used to simulate an airbag. This work also predicts first-order estimates of gas-phase species exit concentrations and characteristic residence times in the inflator. Carbon monoxide, produced as a product of combustion from the high flame temperature propellant, is

partially converted to CO_2 as it flows from the internal combustion chamber to the pressurized plenum before being discharged into the airbag. Specifically, the production/destruction of CO is tracked using three different gas-phase reaction models: chemically frozen, local equilibrium and finite-rate elementary kinetics. The results unambiguously express the need for an airbag combustion program that includes finite-rate, gas-phase kinetics. Finally, the results from the finite-rate CO chemistry model seem in qualitative agreement with the experimental results reported in the literature.

Computer codes that are currently used to simulate airbag inflator performance address almost exclusively the internal performance of airbag inflators. Chemical equilibrium is assumed to determine the products of combustion and flame temperatures. Since the gas generation processes are extremely rapid and over in such a short duration, chemical equilibrium may not be reached, and simplified or detailed chemical kinetics should be considered in the future. In addition, the interaction of the exhaust gas from the gas generator with the ambience has to be taken into account. Addressing the above mentioned issues could allow to use these computer codes to evaluate the performance of SPGG's.

4. Literature Review

As mentioned before, a common way to increase in a controlled way the residence time is by creating a recirculation zone. If the recirculation zone is established at the flame location a "worst case" fire scenario is effectively created. Recirculation zones can be created in many ways and different parameters, including the flame, can affect its size and stability. A literature survey was conducted to gather relevant information that will lead to the definition of the most effective recirculation zone for the purpose of screening SPGG's.

a. Structure of the Flow-Field Behind a Step

A review of previous experimental, numerical and theoretical studies will be presented. The flow behind a step corresponds to a typical reattachment process. It is

worth noting that the reattachment of a shear layer is an important process in a large number of practical engineering configurations including, diffusers, buildings, airfoils, flame stabilization in engines and combustors. Among two-dimensional flows, the backward-facing step is the simplest reattaching flow. The separation line is straight and fixed at the edge of the step, and there is only one separation zone instead of two, as seen in a flow over an obstacle. In addition, the streamlines are nearly parallel to the wall at the separation point, so the step only has a significant effect downstream of the separation point.

The literature presents a large number of studies on large Reynolds number turbulent flows but only a few papers dealing with the structure of a laminar separation-reattachment region of the flow behind a step. A pioneer paper is due to Acrivos and al. [28] who presented results for the steady separated flow past a variety of bluff objects. Among the results there is a pressure distribution along the base of a backward-facing step. Goldstein et al. [29] conducted an experimental investigation of the laminar flow of air over a downstream-facing step. They reported results over a range of 0.36 - 1.02 cm in step height and 0.61 - 2.44 m/s in free stream velocity at the step. The main observation was that laminar reattachment length is not a constant number of step heights as for turbulent flow, but varies with Reynolds number and boundary layer thickness at the step. The shape of the velocity profile at reattachment is found to be similar to the shape of the velocity profile at separation and downstream of reattachment.

Bradshaw and Wong [30] provided the first review of the experimental data for reattaching flows. The review includes experiments on the low-speed flow downstream of steps and fences and especially downstream of a backward facing step. They demonstrate the complicated nature of the flow in the reattachment region and its effect on the slow non-monotonic return of the shear layer to the ordinary boundary-layer state. A key feature of the flow observed by Bradshaw and Wong was the splitting of the shear layer at reattachment, where part of the flow is deflected upstream into the re-circulating flow region to supply the entrainment. Experiments conducted with low velocity turbulent flow [31] showed that at reattachment the re-circulating flow differs from the plane-mixing layer in one important aspect: the flow on the entrainment side is highly turbulent,

as opposed to the low turbulence-level downstream of the reattachment point. The re-circulating flow cannot be characterized as a dead air zone. The maximum measured back-flow velocity is usually over 20% of the free stream velocity and very large negative friction coefficients are observed [32]. The flow in this region is very unsteady with very large turbulent structures (at least as large as the step height) passing through the reattachment region [33,34]. These observations are of great importance in the present application since it ensures the entrainment of air to the re-circulation zone, adequate mixing and the evacuation of combustion products.

The unsteady nature of the flow has been described by means of flow visualization which showed that the length of the separation region fluctuated, therefore the impingement location of the shear layer moved up and downstream. Quantitative measurements confirmed this conclusion and showed that the short-time averaged reattachment point deviated from the long-time averaged reattachment location by as much as ± 2 step heights. A non-dimensional frequency for this motion (\bar{f}) was given by the following expression

$$\bar{f} = \frac{f \cdot X_R}{U_0}$$

where f is the fluctuation frequency X_R is the time averaged distance between the reattachment location and the step and U_0 the inviscid-flow velocity upstream of the step. The characteristics of the flow seemed to be governed by this non-dimensional parameter. Mulin and al. [35] reported that for $\bar{f} < 0.5$ the recirculation appeared to wash out periodically. For $\bar{f} > 0.5$ the recirculation zone appeared stable but smaller than for the steady free-stream case. Further experimental results that tracked the maximum energy content of wall pressure fluctuations [36] validated this observation.

Flow reversal in the reattachment region appears to occur in a random manner with no apparent correlation between the near-wall flow upstream and downstream of reattachment. The largest structure in the flow originates from the roll-up and multiple pairing of span wise vortices [37]. This roll-up is similar to the vortex roll-up and a

pairing process is seen in the free-shear layer. The convective speed of these structures is about $0.6U_o$. The span wise coherence or organization of these vortical structures starts to break down about 3 step heights downstream of detachment with the turbulence structure becoming fully three-dimensional upstream of reattachment. The turbulence intensity of the detached flow is 5-10% higher than for the plane-mixing layers. This is believed to be the result of a very low frequency ($\bar{f} < 0.1$) vertical or flapping motion of the reattaching shear layer.

The vertical or flapping motion of the reattaching shear layer encourages the believe that this kind of detached flow can be manipulated by imposed unsteadiness. Reisenhnel and al. [38] used an oscillating flap of length “h” at the backstep. At low frequencies ($k = f \cdot h / U_o = 0.01-0.05$) and small flap extensions (45°), the flap does not generate much vorticity but periodically blocks the accumulation of vortical fluid at the backstep. This causes the shear-layer vortices to be drawn closer to the plate, thus shortening the reattachment length. For $k = 0.06-0.09$ and full flap extension (90°), the flap tip produces significant vorticity that merges with the shear layer to reduce X_R by 50%. Beyond $k=0.1$ with a full flap extension, the flap vortices are so close together that they do not allow the separated flow to reattach until they have diffused downstream, which produces a larger X_R .

The impingement point of the shear layer moves up and downstream as a function of time [32,33,34] but, average values have been commonly used to characterize the reattachment zone. The most important dependent parameter characterizing the flow field seems to be the average reattachment length which varies from 0 to more than 20 step heights (Figure 1). Comparison of the reattachment length from various experiments provides insight into the effects of varying the following six principal independent parameters:

1. The initial state of the flow. As an example Figure 1 shows the differences between a parabolic and a fully developed velocity profile immediately upstream of the step (Ref.[41]-p) and a developing velocity profile [Ref. [41]). As the flow becomes fully turbulent, $Re_s > 6000$, (Ref. [39]) the length of the recirculation zone becomes independent of the Reynolds number and other recirculation patterns appear at the top

and bottom of the channel [39]. According to Bradshaw [42], recirculation zones originating from laminar boundary layers grow more rapidly than those originating from turbulent boundary layers.

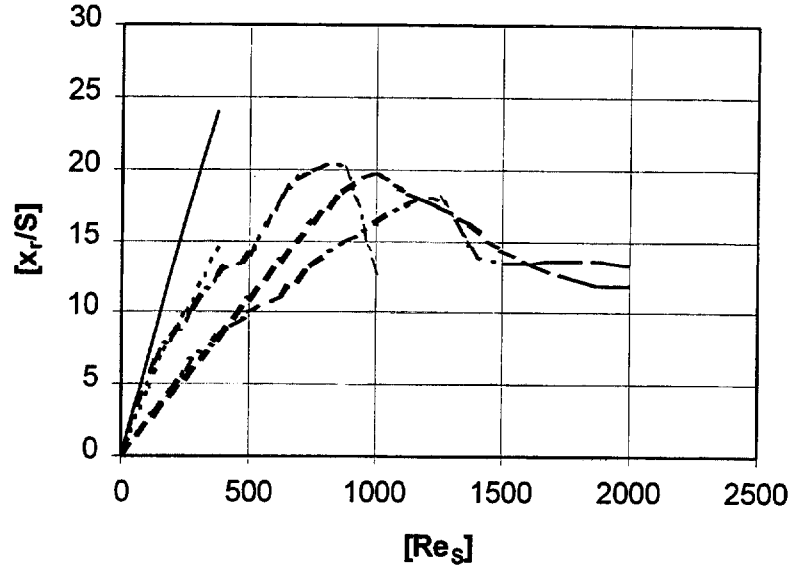


Figure 1. Normalized size of the recirculation zone as a function of the Reynolds number

2. The pressure gradient can have a significant effect on the reattachment length [43].
3. The aspect ratio of the test section has a negligible effect on the re-attachment length for aspect ratios greater than ten [44]. Below ten the reattachment length increases for a laminar boundary-layer and decreases for a turbulent flow, the difference is apparently due to the effects of the corner flow. Figure 1 shows experimental results for an aspect ratio of 1.2 (Ref.[40]) and aspect ratio of 10 (Ref. [39]) showing the above mentioned differences.
4. Blowing or suction tunnels behave in a very similar way.
5. Blowing immediately downstream of the step (for flame attachment purposes) was studied by Richardson et al [45] in a section with aspect ratios of six and four, flow velocities between 50 and 80 m/s and injection velocities ranging from 0 to 0.5 m/s. It was observed that the recirculation zone changes in size and injection had a de-

stabilizing effect. These results correspond to an entirely different velocity regime but the results will be considered when defining the present study.

6. The Reynolds number. At very low Reynolds numbers, the flow is viscous, and the recirculation zone is formed of one stationary eddy at the corner of the step. As the Reynolds number increases, an eddy may detach and decay as it moves downstream. At moderate values, the flow reaches a state of transition, with eddy shedding at the step. At high Reynolds numbers, the flow exhibits turbulent behavior with a continuous process of eddy formation and pairing. The length of the recirculation increases with Reynolds number, reaching a maximum at transition and then decays to a shorter length at the turbulent range. Turbulence statistics show a rise at transition (Figure 1). Flow visualization showed that the length of the separation region fluctuated so that the impingement location of the shear layer moved up and downstream [30].

Numerical calculations incorporating the k - ϵ and algebraic-stress model terribly overpredict experimental observations and especially the turbulence dissipation rate [46]. The numerical study of Amano and Goel [47], using the same principles showed similar problems. In both cases the recirculating region (size and characteristics) is poorly described. Miao and al. [48] indicated that a standard k - ϵ model can yield good predictions for the back-step problem only when modified with an independently calibrated anisotropic eddy viscosity. Preliminary computations conducted using the methodology described by McGrattan et al [9], on a one aspect ratio section, show qualitative agreement with the experimental data (Figure 1 [Model]), further exploration of the use of this model needs to be done and is part of the present proposal. The results presented in Figure 1 do not correspond to the average length of the recirculation zone but to the arithmetic average between the maximum and minimum transient values, therefore the discrepancy between the model and the experimental results still requires further analysis. Quantitative measurements have shown that the short-time average reattachment differed from the long-time average reattachment location by as much as ± 2 step heights.

Many of the papers above include comparative studies where the backward facing step was substituted by either a baffle or a trench. Other studies include sudden expansions and contractions. The qualitative observations presented above seem to be general to all configurations and quantitative details are Acrivos et al [28] Bradshaw and Wong [30] and Eaton and Johnston [31]. Emphasis has been given to the backward facing step because among two-dimensional flows this is the simplest form of reattaching flow.

b. Heat Transfer Characteristics Behind a Step

Heat transfer in the separation and reattachment zones has been the subject of a very reduced number of studies [49-58]. Earlier work by Seban [57] experimentally obtained local heat-transfer coefficients as a function of the flow velocity. Wall temperature distributions in the separated and reattached regions of the flow were also presented. It was found that neither region presented similarity patterns common to flows that are dominated by the friction at the wall. The effect of suction or injection through a slot at the base of the step was also studied indicating small effects on both the pressure distribution and the local heat-transfer coefficient. More recent work by Richardson et al. [45] has shown contradictory results.

Filetti and Kays [56] presented experimental data for local heat transfer rates near the entrance of a flat duct in which there is an abrupt symmetrical enlargement in flow cross section. Although the Reynolds number used for this experiments $70,000 < Re_s < 205,000$) do not correspond to the present application, the results are of interest in that they show that maximum heat transfer occurs at the point of reattachment, followed by a decay toward the values for fully developed duct flow. Empirical equations are obtained for the Nusselt number at the reattachment point as a function of the Reynolds number.

The only work in low speed wind tunnel is that of Aung [58] who presented experiments conducted with a contraction ratio of 12.5/1. Temperature distributions in the gas were measured using a Mach-Zehnder interferometer. Heat transfer upstream of the step is shown to be strongly enhanced by streamline curvature. Downstream of the step the heat transfer increases monotonically in the streamwise direction but is always less than the flat-plate value. Shishov and al. [49] developed an approximate calculation

method for predicting velocity and temperature profiles, reattachment length and local friction and heat transfer coefficients. Their derived analytical expression for heat transfer in the reattachment region correlated well experimental heat transfer data for different types of flow with a sudden expansion.

The work of Yogesh and Raghunandan [50] is devoted to the study of the flow structure and heat transfer characteristics behind a diaphragm in the presence of a diffusion flame. The diffusion flame is stabilized by injecting gaseous fuel into a main stream of air or N_2 - O_2 mixtures. When a diaphragm obstructs the flow over a surface, the flow gets separated at a point upstream of the diaphragm and gets reattached further downstream. Consequently, there are two recirculation zones one in the front and the other behind the diaphragm. The emphasis of this paper is, mainly, on the nature and effects of the downstream recirculation. This attempt to investigate the complex flow resulting from combustion of a fuel surface downstream of a diaphragm has led to some important results concerning large scale features such as heat transfer rates to the surface. Finer features related to the flow pattern and temperature distribution were obtained by particle tracking and fine thermocouples. Blowing or fuel injection through the porous plate tends to lift and stretch the recirculation zone. The flame is located at the shear layer and the eddies appear to be further enlarged. Temperature profiles indicate the expected well-stirred type of regions behind the diaphragm with the surface gradient temperature gradually increasing along the flow direction. The heat flux to the surface shows a constant flux zone near the diaphragm followed by a monotonic rise. The two zones can be attributed to the two different eddies visualized by particle track photographs. Experiments with a heated main stream with no combustion and the flow visualization in the cold flow lead to the conclusion that the flow pattern and heat transfer trends are similar with or without combustion reactions. The presence of a diaphragm helps in postponing the blow-off of the flame in relation to the boundary layer diffusion flame. However, the stable flame zone marked by a perfect two-dimensional flame is only marginally widened. In the boundary layer case, due to the proximity of the flame, the heat transfer rates are high around the leading edge but sharply decrease along the flow direction.

c. *Ignition of a Flammable Mixture Over a Hot Plate*

It is not the objective of this section to summarize the extensive literature on the subject of ignition of a flammable mixture but just to present the main parameters that need to be considered when studying this process. Most of the work on this subject has been conducted by using boundary layer flow and has been summarized in detail by Law and Law [59] and Trevino and Sen [60]. Theoretical work has mostly used asymptotic analysis (high and low activation energy) to determine the parameters that affect the ignition process. The following parameters have been found to affect determine ignition:

- Critical Damköhler number; characteristic residence (flow characteristics) time and chemical times (fuel and oxidizer, fuel and oxydizer concentrations, gas pahase temperature, etc.) determine a critical Damköhler number that needs to be attained for ignition to occur. The attainment of the critical Damköhler number will determine the location of the ignition point. The fuel concentration in the flow is highly dependent on the pyrolysis rate of the fuel and its nature (charring, non-charring).
- The thermal properties of the flammable mixture.
- The thermal properties of the plate; Trevino and Fernandez-Pello [61] show that the thermal properties of the material influenced the hot plate influenced ignition. An isothermal wall can be considered as the best case scenario for ignition, the thermal properties of the wall will indicate how restrictive heat transfer from the wall to the flammable mixture is and subsequently how it deviates from the ideal case.
- Catalytic reactions at the surface; many papers have dealt with the effect of catalytic reactions at the hot plate surface. The characteristic chemical time of the catalytic reactions is the main parameter that quantifies the importance of the chemical interaction between the solid and gas phases. Slow catalytic reactions have negligible effect on ignition, fast catalytic reactions have a dominant effect (favorable or adverse) on ignition [62].

5. Diagnostic Techniques

The specific constraints of the present application result in a reduced number of diagnostic techniques that have been proven to provide information that could be useful in the characterization of the controlling mechanisms of extinction by means of SPGG's. This summary only includes those techniques that have been used in studies related to reattaching flows and concentrates on techniques that provide information on the structure and global properties of the flow. Determination of the diagnostic techniques that will determine species concentrations would be of significance but goes beyond the scope of this review.

a. *Velocity Measurements*

Different diagnostic techniques have been used to characterize the flow field behind a backwards facing step. The general criteria for the determination of the adequate diagnostic technique has been the Reynolds number. For the low Reynolds number regime ($Re_s < 2,000$) flow visualization by means of illumination of particles introduced in the stream has proven to provide best results. Finaish and al. [63] successfully investigated the development of two-dimensional vortex patterns in an accelerating flow by means of streak-line visualization. Liu et al. [64] used Particle Image Velocimetry (PIV) to study the transient behavior of a turbulent ($Re=2,872$) flow inside a channel of aspect ratio of 12. Bandyopadhyay [65] studied the nature of instabilities and large structures generated in reattaching boundary layers and especially behind a backward-facing step by the simultaneous use flow visualization and hot-wire anemometry. Plane-flow visualization was conducted in the backward-facing step showing that the three-dimensional mixing layer developing after detachment persists even after reattachment. Velocity measurements obtained by means of both techniques proved to be identical with flow visualization having the advantage of providing a flow field instead of a single measurement. Honji [66] investigated the incompressible starting flow past a downstream-facing right-angled step at $Re_s < 500$ by means of flow-visualization

techniques. The starting flow down a step height of 1.85 cm was characterized by the Reynolds number based on the step height ($Re_s = U.S/\nu$) and the dimensionless time, ($\tau = U.t/S$). The formation of secondary vortices, in the recirculation zone, for $Re_s > 120$ and a third vortex for $Re_s > 140$ was characterized by means of this technique. In a similar way, Sinha and al.[40] reported the results of an experimental investigation carried out at a wind speed of 1.8m/s over 3 backward facing steps 0.625, 1.25, and 2.5cm in height. In all cases the separating boundary layer was laminar and 1.4cm thick. Data was obtained by means of smoke visualization and hot wire measurements. The data served to further characterize the reattachment length in the Reynolds number range 100 – 12,500 and to describe the general patterns of the flow between the point of separation and that of reattachment.

An important consideration for the success of PIV measurements is determined by type of material used for seeding the flow. The constraints placed on the seeding material are: 1) the particle's inertia must be small enough that it faithfully follows the motion of the fluid, 2) the particles settling velocity must be small in comparison to the typical velocities of the flow, and 3) the particle must be large enough so that it will provide sufficient scattered light intensity to register on the CCD [67,68]. The best candidates for the current application would be fine hollow glass beads ($\sim 2 \mu\text{m}$ diameter) or Titanium Oxide (TiO_2) power ($\sim 4 \mu\text{m}$) introduced through an aerated bed. The suitability of these particles for the imaging conditions above can be checked by examining hydrodynamic response of the particles to oscillatory forcing, checking the settling velocity, and estimating the expected scattered light intensity from the particles.

The response of a particle to unsteady motion has been calculated by Hjelmflet and Mockros [69], and for large particle to fluid density ratios ($\rho_p / \rho_f \ll 1$) they found the expression

$$\frac{u_p}{u_f} = \frac{1}{\sqrt{1 + (2\pi St)^2}}$$

where $St = D^2 \rho_p f / 18 \rho_f \nu$ is the Stokes number and is equivalent to the ratio of the particle viscous response time to the frequency of fluctuation of the flow, f . The

calculated frequency response (where $u_p/u_f = 0.99$) for the glass beads and the TiO_2 is then found to be approximately 2300 Hz and 115 Hz, respectively. These values are in excess of any unsteady fluid motion expected for the recirculating flows of interest.

The settling velocity can be estimated using a simple balance between the Stokes drag force and buoyancy, giving

$$v_{\text{settle}} = \frac{D^2 \rho_p g}{18 \rho_f \nu}$$

The calculated settling velocities correspond to 0.01 cm/s for the glass beads and 0.2 cm/s for the TiO_2 , both of which are small compared to the expected mean velocities of 0.1 – 1.0 m/s.

The last quantity of concern is whether the particles scatter enough light to be detected by the camera. Adrian and Yao [70] have made calculations of mean exposure level generated by spherical particles under different imaging conditions, using the equation

$$\bar{\epsilon} = \frac{\lambda}{\pi^3 (M^2 D^2 + d_s^2)} \frac{W}{\Delta y \Delta z} \int \sigma^2 d\Omega$$

where λ is the wavelength of the light, M is the magnification, d_s is the diffraction limited point-response diameter of the lens, $W = I_0 \delta t$ is the energy of the incident laser pulse, Δy is the width of the light sheet, Δz is the light sheet thickness, σ is the Mie scattering coefficient, and Ω is the solid angle subtended by the imaging lens. The most stringent conditions will occur for the smallest particles at large $f\#$, and using Adrian and Yao's calculations (see their figure 6) give a non-dimensional exposure intensity of $\bar{\epsilon} \Delta y \Delta z / W = 7 \times 10^{-9}$ for $D = 2 \mu\text{m}$ and $f\# = 22$. For the 2 W laser formed into a 10 cm \times 0.1 cm light sheet used in the proposed experiments, this gives a dimensional irradiance of 3.5 nW/cm², which is 35 times greater than the minimum irradiance required for the camera (0.1 nW/cm). Further gains in light intensity can be made by lowering the $f\#$ several stops.

b. Time Dependent Density Measurements

The use of Mach-Zender Interferometry, Schlieren Deflectometry and Rainbow Schlieren Deflectometry is documented in the literature. The results obtained are generally of qualitative nature. Rainbow Schlieren Deflectometry offers significant advantages over traditional knife-edge-Schlieren in that the eye is much more sensitive to color changes than intensity variations, and there are now techniques to quantify the refractive index field. Rainbow Schlieren permits a quantitative determination of temperature gradients in the gas phase, temperature gradients in the test section will deflect the beam and cause it to pass through different colors on the filter which will be recorded in the image on the camera. Depending on the orientation of the color filter, either horizontal or vertical gradients are measured.

The basic equation governing the deflection of a light ray passing through a temperature gradient is:

$$\delta_y = \frac{fL}{n_0} \frac{\partial n}{\partial T} \frac{\partial T}{\partial y}$$

where δ_y is the deflection at the Schlieren filter plane, f is the focal length of the decollimating mirror, L is the path length through the medium, $\partial n/\partial T$ is the change in index of refraction with temperature, with n_0 the reference index of refraction, and $\partial T/\partial y$ is the temperature gradient. Temperature gradients will result, therefore, in beam deflections that will be recorded by the camera.

c. Heat Transfer to the Walls

Standard Schmidt-Boelter type thermopiles together with surface thermocouple measurements have been used in most of the studies that deal with heat transfer at the walls and that are reviewed above [29-58]. The thermopiles provide linear millivolt output directly proportional to the heat flux through the gage. These gages have relatively fast response times (order .3 seconds) and are small in size (3.13 mm in diameter). They can operate in relatively hot environments (calibrated to over 473 K) and can be subjected to heat fluxes as large as 0.3 MW/m². The response time and robustness of the device seem to be appropriate for the present application.

6. Characteristics of the Poitiers' Intermediate-Scale Fixture

A study on cross-flow effects over a pool fire has been recently developed at Poitiers and the main results are presented in references [71-76]. They mainly concerns have been the effects of confinement and cross flow on an intermediate scale pool-like diffusion flame. Especially the case where the fire is confined in a tunnel of similar characteristic length scale as that of the flame. The geometrical characteristics of the flame have been determined for different cross-flow velocities, fuel injection surface and velocity.

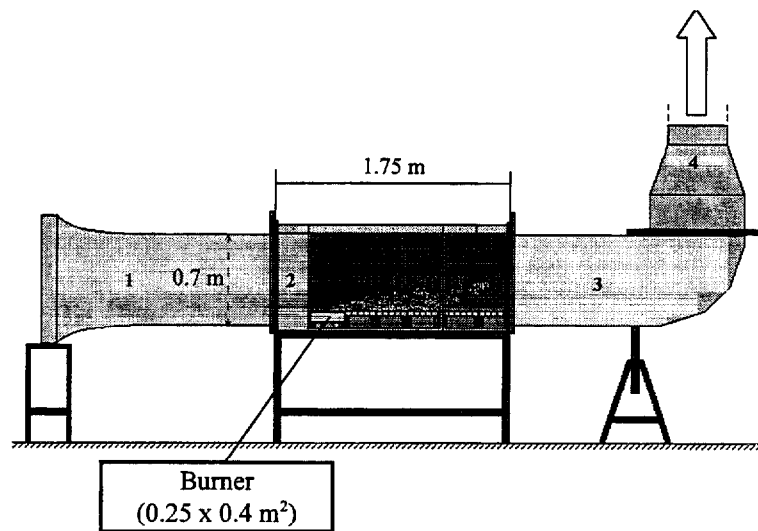


Figure 2 – Experimental apparatus at the University of Poitiers

A schematic of the diagram of the wind-tunnel is presented in figure 2. The experimental apparatus consist of a 5.25 m long horizontal channel through which air motion was induced by means of a centrifuge fan. The center part of the wind tunnel is the test section, 1.75 m long, 0.7 m height and 0.4 m wide (2 on figure 2). The walls, floor and ceiling are each made of seven removable actively cooled panels (0.7 m x 0.4 m for the walls and 0.4 m x 0.25 m for floor and ceiling). The wall panels can be substituted by windows, for observation, and the floor panels by burners. The burners covers the entire cross section of the tunnel. Each burner consists of a rectangular sintered bronze

slab (20 mm in thickness) with imbedded copper cooling tube (8 mm in diameter). The thickness of the slab and the diameter are chosen to guarantee homogeneous flow through the entire burner surface. The burner width can be changed from as small as desired up to cover the entire channel width (0.4 m). The burner length can be varied in a discrete way, 0.25 m, 0.50m, and 0.75 m, being the length the stream wise direction. Water from a reservoir at constant temperature of 65°C was pumped through the cooper tube to keep the burner surface at a given temperature and to avoid condensation. Similar burners have been successfully used in the past to simulate the pool or wall fire combustion of a liquid or of a solid fuels [71-73, 77, 78].

The characteristic velocity of the main flow can be obtained by means of a screw gauge placed in the symmetry plane of the test section or by Laser Doppler Velocimetry measurements. The wind tunnel can operate on both, blowing and aspiration modes. The rotating frequency of the fan is regulated to keep the flow velocity constant through out an experiment. In the aspiration configuration the air is conducted into the test section through a convergent (1 on figure 2), 1.8 m long, with a restriction ratio of 3.57. To homogenize the flow, a honeycomb, 50 mm in thickness, is fixed at the entry of the convergent. Combustion products are driven away through a collector (3 on figure 2), 1.7 m long. To minimize environmental perturbations present in the laboratory, the floor of the channel is placed 0.9 m from the room floor. A similar arrangement is used when in the blowing mode.

An initial aerodynamic study of the wind tunnel pointed out that the best method to induce a fully developed velocity profile upstream of the burner is by means of aspiration. Aspiration and blowing experiments were also conducted with flames and showed no significant differences. The geometry was chosen to guarantee, keeping minimum dimensions, that the flow will reproduce a turbulence intensity (u'/\bar{U}) typical of fire scenario (between 5% and 15%) [72].

Temperature measurements can be performed using a combination of 50 μm type K thermocouples and video recordings. A CCD camera records the visual characteristics of the flame on a U-matic video recorder. The image processing used to extract the

geometric parameters of the flame (length, height, angle of inclination, etc...) is a refinement of the technique extensively described in references [71-73].

Three parameters can easily be varied to performed a parametric study: the fuel, the cross flow velocity (U_∞), the fuel injection velocity (V_f), the the burner lenth (X_b) and width (Y_b). The common range for the air flow velocity (U_∞) is between 0.5 and 2.5 m/s. The fuel injection velocity is restricted by the source Froude number which, for fire flames, should be kept in the range of 10^{-4} - 10^{-6} . The values used for the fuel injection are then between 1.9 and 5 mm/s. If the fuel is propane characteristic injection velocities will provide theoretical heat release rate is ranging between 18 kW and 135 kW.

An example of the use of this experimental facility is presented in figure 3. The figure presents as presence probability distribution obtained from digitalization of a top view of the flame (figure 3(a)). From this image a mean flame is defined as where the presence probability is 50% (figure 3(b)). The zone where the intermittence is 50% is finite, therefore to obtain the main flame border the arithmetic average of the internal and external boundaries of the 50% intermittence zone is taken and define as the main flame border.

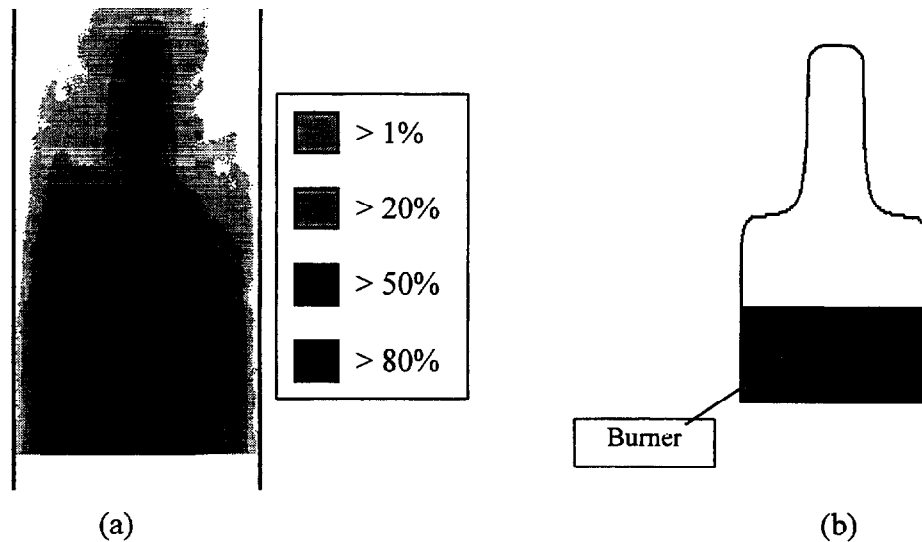


Figure 3 –(a) Presence probability of the flame in the transversal plane. (b) Mean flame contour.

It is not the intention of the present report to describe the experimental results obtained with this facility and that are described in detail in references [71-76]. The objective is to present the facility and its potential use for controlled intermediate scale validation of the small scale results derived from the bench scale screening test.

Placing a simulated rectangular pool burner inside the horizontal channel of the wind tunnel describe above, it has been possible to efficiently manage air entrainment into the flame and to reproduce different levels of confinement. It has been underlined that the total mass flow generated by the flame is independent of the level of confinement and the flame height is significantly affected by a preferential orientation of the air entrained. Simple modifications to this facility will provide a “worst case” intermediate fire that follows well the philosophy driving the design of the bench scale screening test for SPGG’s, thus, provide a validation mechanism at a more realistic length scale.

7. Numerical Methodology

As mentioned before, the methodology proposed by McGrattan and al. [9] was used in the present program. The main characteristics of their approach to field modeling fire phenomena which emphasizes high spatial resolution and efficient flow solving techniques are presented in the above mentioned reference. This approach has been fostered by the ever-increasing power of computers and the development of faster numerical algorithms. To make the most of the current generation workstations, they have focused their efforts on developing relatively simple numerical algorithms to address the transport of combustion products in relatively simple enclosures. This approach will enable the use of a PC-Pentium computer for the computations.

The mixing and transport of combustion products is calculated directly from an approximate form of the Navier-Stokes equations. This approximation involves the filtering out of acoustic waves while allowing for large variations in temperature and density [79]. This gives the equations a highly elliptic character, consistent with low speed, thermal convective process. In fact, a Poisson equation is solved at each time step in the calculation with an FFT-based direct solver [80].

The fire itself is prescribed in a manner consistent with a mixture fraction based approach to combustion, but the combustion themselves are not, up to now, simulated directly and sub-grid scale motions are treated following the analysis of Smagorinsky [81]. The extent to which the fire is resolved depends on the objective of the calculation. As shown in the literature review, in a recirculating zone, such as the one of interest in this program, the reacting zone can be modeled simply as a source of heat and mass. The heat release and species production rates would be determined from experiment.

Present capabilities permit fully three dimensional simulations at about 2 mm resolution for the proposed experimental facility. The products are simulated by tracking a large number of Lagrangian elements that originate reaction zone. These same elements carry the heat released by the fire, providing a self consistent description of the product transport at all resolvable length and time scales. Large temperature and pressure variations are permitted, subject to the limitation that the Mach number is much less than one. The methodology has already been successfully used to study a number of fire-related phenomena as shown in references [82], [83], and [84].

The LES code has already been used to perform successful preliminary isothermal flow calculations in two and three dimensional configurations similar to the proposed geometry (see Figure 1). It needs to be noted that the treatment given to the viscosity by the Smagorinsky [81] model results in an erroneous velocity determinations for a potential flow. In contrast, for a fully developed entry profile where viscous stresses are present the results are satisfactory. A simple correction for the zero gradient limit should be able to solve this problem.

8. Conclusions

The literature review indicates clearly that it is possible to well describe the structure of a recirculation zone located behind a backward facing step and consequently to design a fixture with which will allow to establish a diffusion flame stable over a large range of experimental conditions. Moreover, one can obtain the heat transfer profile at the burning surface and then, localize the hot spots where re-ignition would be favored.

The following issues need to be taken into consideration when proceeding to the design of a test fixture around the recirculation zone induced by a backward facing step, baffle or trench:

- Mixing of fuel and oxidizer is guaranteed since splitting of the shear layer at reattachment results in part of the flow being deflected upstream into the recirculating flow region to supply the oxidizer entrainment into the fuel zone.
- A significant increase in mixing occurs inside the recirculation zone since the flow on the entrainment side is highly turbulent, as opposed to the low turbulence-level downstream of the reattachment point. Care has to be taken not to increase the strain rate inside the recirculation zone to levels where extinction is induced by the main flow field. Systematic study of the operation conditions needs to be conducted as a function of the fuel to be used.
- The maximum measured back-flow velocity is usually over 20% of the free stream velocity. This value sets a criteria for fuel injection to the recirculation zone to guarantee optimal burning conditions.
- The flow in this region is very unsteady with very large turbulent structures (at least as large as the step height) passing through the reattachment region. This observation is of great importance in the present application since it ensures the entrainment of air to the re-circulation zone, adequate mixing and the evacuation of combustion products. The transient evolution of the recirculation zone needs to be further evaluated for the conditions typical of the present application. The choice of a specific geometry such as backward facing step a baffle or a trench will have a

significant effect on the stability of the reaction zone. Stability of the recirculation zone results in low entrainment and low product evacuation, thus, not necessarily in a more stable reaction zone.

- The aspect ratio of the test section has a negligible effect on the re-attachment length for aspect ratios greater than 10. For values smaller than 10 the reattachment length increases for a laminar boundary-layer and decreases for a turbulent flow.
- Upstream perturbations can play an important role on the characteristics and stability of the eddy behind the step or obstacle.
- Blowing or suction tunnels behave in a very similar way.
- At least for high free stream velocities, injection has a de-stabilizing effect. No experimental data was found for velocities in the range of interest.
- The length of the recirculation increases with Reynolds number, reaching a maximum at transition and then decay to a shorter length at the turbulent range. For $Re > 1500$ the recirculation zone has an almost constant length. For $Re > 6000$ other recirculation patterns appear at the top and bottom of the channel increasing the complexity of the flow.

Numerical calculations incorporating the $k-\epsilon$ and algebraic-stress model over predict experimental observations and especially the turbulence dissipation rate. Preliminary computations conducted using the methodology described by McGrattan et al, on a one aspect ratio section, show qualitative agreement with the experimental data, further exploration of the use of this model needs to be done.

For the low Reynolds number regime ($Re_s < 2,000$) flow visualization by means of illumination of particles introduced in the stream has proven to provide best results. Particle Image Velocimetry is recommended as an appropriate technique to resolve the flow field characteristics. For punctual measurements of the turbulence intensity Laser Doppler Velocimetry will be necessary. Recommendations of seeding particle size and type are provided in the text of the report. Classic temperature and heat flux measurements seem adequate for the present objectives.

The test facility in Poitiers (France) seems well suited for future validation, at a more realistic length scale, of the screening procedure for SPGG's. The information provided on the capabilities of this test facility can be used as background information on the design process. It also shows the potential of flow visualization techniques, velocity, turbulence and temperature measurements as tools to characterize air entrainment and flame structure.

9. References

1. Proceedings of the Halon Alternatives Technical Working Conference, University of New Mexico, Albuquerque, May 11-13, 1993.
2. Dinunno, P.J., "*Perspective on halon replacements and alternatives. Keynote*," Proceedings of the Halon Alternatives Technical Working Conference, University of New Mexico, Albuquerque, May 11-13, 1993.
3. Tapscott, R.E., "*Halon substitutes: An overview*," Proceedings of the Halon Alternatives Technical Working Conference, University of New Mexico, Albuquerque, May 11-13, 1993.
4. Grosshandler, W.L., Gann, R.G. and Pitts, W.M., Eds., "*Evaluation of Alternative In-flight Fire Suppressants for Full-scale Simulated Aircraft Engine Nacelles and Dry Bays*," **NIST SP 861**, National Institute of Standards and Technology, Gaithersburg, MD, April 1994.
5. Grosshandler, W.L. and Gann, R.G., "*Low Environmental Impact Fire Suppression Concepts*," (unpublished note), 1995.
6. Yang, C.Y. and Grosshandler, W.L., Eds., "*Solid Propellant Gas Generators: Proceedings of the 1995 Workshop*," **NISTIR 5766**, National Institute of Standards and Technology, Building and Fire Research Laboratory, Gaithersburg, MD 20899, 1995.
7. Grosshandler, W.L., Yang, Y.C. and Cleary, T.G. "*Screening Methods for New Fire Suppression Technologies*," Proceedings of the 1996 International Conference on Ozone Protection Technology, Washington D.C., October 21-23, 1996.
8. Hamins, A. "*Aspects of Flame Suppression*," **NISTIR 5766**, National Institute of Standards and Technology, Building and Fire Research Laboratory, Gaithersburg, MD 20899, 1995.
9. McGrattan, K.B., Baum, H.R. and Rehm, R.G., "*Large eddy simulation of fire phenomena*," NIST internal report, Gaithersburg, MD 20899, august 12, 1997.
10. McGrattan, K.B., Rehm, R.G. and Baum, H.R., "*Fire driven flows in enclosures*," *Journal of Computational Physics*, **Vol.110**, n° 2, p.285, 1994.

11. Williams, F.A. "*Combustion Theory*," The Benjamin Cummings Publishing Company, Inc., 2nd Edition, 1985.
12. Blazowski, W.S., Progress in Energy and Combustion Science, **4**, 177, 1978.
13. Carrier, G., Fendell, F., Feldman, P. "*Forced-convection extinction of the diffusion flame supported by a pyrolyzing body*," FEMA Report Contract DCPA01-78-C-0325, October 1979.
14. Carrier, G. Fendell, F., Feldman, P. and Fink, S., "*Forced convection extinction of a diffusion flame sustained by a charring body*," Combustion Science and Technology, **Vol.28**, pp.271-304, 1982.
15. Hamins, A., Cleary, T., Borthwick, P., Gorchkov, N., McGrattan, K., Forney, G., Grosshandler, W.L., Presser, G., Melton, L. "*Suppression of Engine Nacelle Fires*," Fire Suppression System Performance of Alternative Agents in Aircraft Engine and Dry Bay Laboratory Simulations, NIST SP 890, Vol.II, Chapter 9, Gann, R.G. Editor, NIST, November 1995.
16. Ashley, S., "*Automotive safety is in the bag*," Journal of Mechanical Engineering, pp.59-64, January 1994.
17. "*Airbag gas generants side effects studied*," Journal the Society of Automotive Engineers, **Vol.87**, n°6, pp.82-84, 1979.
18. "*Development of a test device for airbag gas generants*," Journal the Society of Automotive Engineers, pp.32-34, June 1992.
19. "*Airbags: education and experience*," Journal the Society of Automotive Engineers, pp.29-32, September, 1993.
20. Berger, J.M. and Butler, P.B., "*Equilibrium analysis of three classes of automotive airbag inflator propellant*," Combust. Sci. and Tech., **Vol.104**, pp.93-114, 1995.
21. Wang, J.T., "*Recent advances in modeling of pyrotechnic inflators for inflatable restraint systems*," ASME Winter Annual Meeting, San Francisco, Ca., **AMD-Vol.106**, pp.89-93, December 10-15, 1989.
22. Motevalli, V. and Bedewi, N.E., "*Airbag modelling, An overview of airbag deployment modelling*," IFMS 95, pp.1-11, Coco Beach, Fl., March 13-15, 1995.

23. Lupker, H.A. and Bruijs, W.E.M. "*Gas jet model for airbag inflators,*" SAE Technical Paper Series n°.930645, pp.85-94, 1993.
24. Groenenboom, P., Lasry, D., Subbian, Th. and Narwani, G., "*A diffusive gas jet model in PAM-SAFE for airbag inflation,*" SAE Technical Paper Series n°.930238, pp.1-9, 1993.
25. Krier, H., Butler, P.B., "*Modeling and experimental validation of pyrotechnic gas generator, Proceedings of the 1995 workshop: solid propellant gas generators,*" Report NISTIR 5766, BFRL/NIST, pp.89-122, Gaithersburg, Md 20899, USA, June 28-29, 1995.
26. Butler, P.B., Kang, J. and Krier, H., "*Modeling and numerical simulation of the internal thermochemistry of automotive airbag inflators,*" Prog. Energy Combust. Sci. , **Vol.19**, pp.365-382, 1993.
27. Schmitt, R.G., Butler, P.B. and Freesmeier, J.J., "*Performance and CO production of a non-azide airbag propellant in a pre-pressurized gas generator,*" Combust. Sci. and Tech., Vol.122, pp.305-330, 1997.
28. Acrivos, A., Leal, L.G., Snowden, D.D., Pan, F., "*Further experiments on steady separated flows past bluff objects,*" J.Fluid Mech., **34**, Part 1, pp.25-48, 1968.
29. Goldstein, R.J., Erikson, V.L., Olson, R.M. and Eckert, E.R.G., "*Laminar separation reattachment and transition of flow over a downstream-facing step,*" Trans.ASME D, J.Basic.Engng., **92**, pp.732-741, 1970.
30. Bradshaw, P. and Wong, F.Y.F., "*The reattachment and relaxation of a turbulent shear layer,*" J.Fluid Mech.,**52**, part 1, pp 113-135, 1972.
31. Eaton, J.K. and Johnston, J.P., "*A review of research on subsonic turbulent flow reattachment*", AIAA J., **19**, pp.1093-11100, 1981.
32. Eaton, J.K. and Johnston, J.P., "*Turbulent flow reattachment: An experimental study of the flow and structure behind a backward-facing step,*" Report MD-39, Dept.of Mechanical Engineering, Stanford University, 1980.
33. Abbot, D.E. and Kline, S.J. "*Experimental investigations of subsonic turbulent flow over single and double backward facing step,*" Trans.ASME D: J.Basic Engng , **84**, p;317-325, 1962.

34. Kim, J., Kline, S.J. and Johnston, J.P., "*Investigation of separation and reattachment of a turbulent shear layer: Flow over a backward-facing step*," Report MD-37, Thermosciences Div., Dept. of Mechanical Engineering, Stanford University, 1978.
35. Mulin, T., Greated, C.A. and Grant, I. "*Pulsating flow over a step*," Phys. Fluids, Vol.24, pp.669-674, 1980.
36. Driver, D.M., Seegmiller, H.L. and Marvin, J. "*Time dependent behavior of a reattaching shear layer*," AIAA J., Vol.25, pp.914-919, 1987.
37. Pronchick, S.W. and Kline, S.J. "*An experimental investigation of the structure of a turbulent reattaching flow behind a backward facing step*," Report MD-42, Thermosciences Division, Department of Mechanical Engineering, Stanford University, Stanford California, June 1983.
38. Reisenthel, P.H., Nagib, H.M. and Koga, D.J., "*Control of separated flows using forced unsteadiness*," AIAA Paper No 85-0555, 1985.
39. Armaly, B.F., Durst, F., Pereira, J.C.F. and Schonung, B., "*Experimental and theoretical investigation of backward-facing step flow*," J. Fluid Mech., 127, pp.473-496, 1983.
40. Sinha, S.N., Gupta, A.K. and Oberai, M.M., "*Laminar separating flow over backsteps and cavities, Part I: Cavities*," AIAA J., 19, pp.1527-1530, 1981.
41. Denham, M.K. and Patrick, M.A., "*Laminar flow over a down-stream facing step in a two-dimensionnal flow channel*," Trans. Inst. Chem. Engrs. 52, p.361-367, 1974.
42. Bradshaw, P. "*The effect of initial conditions on the development of a free shear layer*," J. Fluid Mech., Vol.26, pp.225-236, 1966.
43. Kuehn, D.M., "*Effects of adverse pressure gradient on the incompressible reattaching flow over a rearward-facing step*," AIAA J., 18, pp.343-344, 1980.
44. DeBrederode, V. and Bradshaw, P., "*Three-dimensional flow in nominally two-dimensionnal separation bubbles. I Flow behind a rearward-facing step*," Imperial College, Aeronautical Report 72-19, 1972.
45. Richardson, J. deGroot, W.A., Jagoda, J.I. Walterick, R.E., Hubbart, J.E. and Strahle, W.C., "*Solid fuel ramjet simulator results: Experiment and analysis in cold flow*," J. Propulsion, 1, pp. 488-493, 1985.

46. Driver, D.M. Lee, H. and Seegmiller, "Features of a reattaching turbulent shear layer in divergent channel flow", AIAA J., **23**, pp.163-171, 1985.
47. Amano, R.S. and Goel, P., "Computations of turbulent flow beyond backward-facing steps using Reynolds-stress closure," AIAA J., **23**, pp.1356-1361, 1985.
48. Miau, J.J. Lee, K.C., Chen, M.H. and Chou, J.H., "Control of separated flow by a two-dimensional oscillating fence," AIAA J., **29**, pp.1140-1148, 1991.
49. Shishov, E.V., Roganov, P.S., Grabarnik, S.I., Zabolotsky, V.P., "Heat transfer in the recirculating region formed by a backward-facing step," Int.J.Heat Mass Transfer, **31**, pp.1557-1562, 1988.
50. Yogesh, G.R. and Raghunandan, B.N., "Flow structure and heat transfer characteristics behind a diaphragm in the presence of a diffusion flame," Int.J.Heat Mass Transfer, **32**, pp.19-28, 1989.
51. Seki, N., Fukusato, S. and Hirata, T., "Effect of stall length on heat transfer in reattached region behind a double step at entrance to an enlarged flat duct," Int.J.Heat Mass Transfer, **19**, pp.700-702, 1976.
52. Lin, J.T., Armaly, B.F. and Chen, T.S. "Mixed convection heat transfer in inclined backward-facing step flows," Int.J.Heat Mass Transfer, **34**, pp.1568-1571, 1991.
53. Mendez, F. Trevino, C. and Linan, A., "Boundary layer separation by a step in surface temperature," Int.J.Heat Mass Transfer, **35**, pp.2725-2738, 1992.
54. Hong, B., Armaly, B.F., Chen, T.S., "Laminar mixed convection in a duct with a backward-facing step: the effects of inclination angle and Prandtl number," Int.J.Heat Mass Transfer, **36**, pp.3059-3067, 1993.
55. Yang, J.T. and Tsai, C.H., "High temperature heat transfer of separated flow over a sudden-expansion with base mass injection," Int.J.Heat Mass Transfer, **39**, pp.2293-2301, 1996.
56. Filetti, W.G. and Kays, W.W., "Heat transfer in separated, reattachment and redevelopment regions behind a double step at the entrance of a flat duct," Trans.ASME C, J.Heat Transfer, **89**, p.163, 1967.

57. Seban, R.A. "Heat transfer to the turbulent separated flow of air downstream of a step in the surface of a plate," Transactions of the ASME, C, J.of Heat Transfer, **86**, pp.259-264, 1964.
58. Aung, W., "An experimental study of laminar heat transfer downstream of back steps," Trans ASME C, J.of Heat Transfer, **105**, pp.823-829, 1983.
59. Law, C.K, and Law, H.K., *Journal of Fluid Mechanics*, 92-97, 1979.
60. Trevino, C. and Sen, M., *Eigteenth Symposium (International) on Combustion, The Combustion Institute*, 1781, 1981.
61. Trevino, C. and Fernandez-Pello, A.C., *Combustion and Flame*, 49, 91-100, 1983.
62. Trevino, C. and Fernandez-Pello, A.C., *Combustion Science and Technology*, 26, 245-251, 1981.
63. Finiash, F. Freymuth, P.F. and Bank, W., "Starting flow over spoilers, double step and cavities," J.Fluid Mech. **168**, pp.383-392, 1986.
64. Liu, Z.C., Landreth, C.C., Adrian, R.J. and Hanratty, T.J., "High Resolution Measurements of Turbulent Structures in a Channel with Particle Image Velocimetry," Experiments in Fluids, 10, 301-312, 1992.
65. Bandyopadhyay, P.R., "Instabilities and large structures in reattaching boundary layers," AIAA J., **29**, pp.1149-1155, 1991.
66. Honji, H., "The starting flow down a step," J.Fluid Mech.,**69**, Part 2, pp.229-240, 1975.
67. Adrian,J., "Particle Imaging Techniques for Experimental Fluid Mechanics," Ann. Rev. Fluid Mech., **23**, p 261-304, 1991.
68. Willert E. and Gharib, M.. "Digital Particle Image Velocimetry." Experiments in Fluids, **10**, p 181-193, 1992.
69. Hjelmfelt, T. and Mockros, L.F.,. "Motion of discrete particles in a turbulent fluid." *Appl. Sci. Res.*, **16**, p 149-161, 1966.
70. J. Adrian, J. and Yao, C.-S., "Pulsed laser technique application to liquid and gaseous flows and the scattering power of seed materials," Applied Optics, **24(1)**, 44-52, 1983.

71. Audoin, L., "*Etude de la structure d'une flamme simulant un incendie de produits industriels. Caractérisation et modélisation de cas réels de feux,*" Thèse de Doctorat de l'Université de Poitiers, Poitiers, 24 Janvier 1995.
72. Kolb, G. "*Etude d'une flamme non-prémélangée caractéristique d'un incendie en présence d'un écoulement forcé,*" Thèse de Doctorat de L'université de Poitiers, Poitiers, 22 Mars 1996.
73. Audoin, L., Kolb, G., Torero, J.L. and Most, J.-M., "*Average centreline temperatures of a buoyant pool fire obtained by image processing of video recording,*" Fire Safety Journal, Vol.24, pp.167-187, 1995.
74. Kolb, G., Audoin, L., Most, J.-M. and Torero, J.L., "*Confinement effects on the mean flame height of a buoyant diffusion flame,*" Spring Technical Meeting of the Central States Section of the Combustion Institute, Point Clear, Mobile, Alabama, april 27-29, 1997.
75. Kolb, G., Most, J.-M. and Torero, J.L., "*Simulated pool fires tilted by wind: flame characteristics and geometrical considerations,*" 1997 ASME National Heat Transfer Conference, Baltimore Md., Vol.3, HTD-Vol.341, pp;19-35, 1997.
76. Kolb, G., Torero, J.L., Most, J.-M. and Joulain, P. "*Cross flow effects on the flame height of an intermediate scale diffusion flame,*" International Symposium on Fire Science and Technology (IFST'97), Paper A-17, Seoul, Korea, November 12-14, 1997.
77. Most, J.-M., Sztal, B. and Delichatsios, M.A., "*Turbulent wall fires-LDV and temperature measurements and implications,*" Proceedings 2nd Int.Symposium on Applications of Laser Anemometry to Fluid Mechanic, Lisboa, Portugal, 1984.
78. Annarumma, M., Most, J.-M. and Joulain, P. "*Velocity and temperature measurements in a bi-dimensional pool fire: influence of a vertical wall close to the fire,*" Proceedings 12th ICDERS, dynamics of deflagration and reactor systems: heterogeneous combustion, Progress in Astronautics and Aeronautics, Vol.132, pp.314-338, 1991.
79. Rehm, R.G. and Baum, H.R. "*The equations of motion for thermally driven, buoyant flows,*" Journal of Research of the NBS, Vol.83, pp.297-308, May-June 1978.

80. McGrattan, K.B., Rehm, R.G. and Baum, H.R., "*Fire driven flows in enclosures,*" Journal of Computational Physics, **Vol.110**, n° 2, p.285, 1994.
81. Smagorinski, J., "*General circulation experiments with the primitive equations. I The basic experiment,*" Monthly Weather Review, **Vol.91**, pp.99-164, 1963.
82. Baum, H.R., McGrattan, K.B. and Rehm, R.G., "*Simulation of smoke plumes from large pool fires,*" 25th Symposium (International) on Combustion, The Combustion Institute, pp.1463-1469, 1994.
83. Mell, W.E., McGrattan, K.B. and Baum, H.R., "*Numerical simulation on combustion in fire plumes,*" 26th Symposium (International) on Combustion, The Combustion Institute, pp.1523-1530, 1996.
84. Baum, H.R., McGrattan, K.B. and Rehm, R.G., "*Three dimensional simulations of fire plume dynamics,*" Jour.HTSJ, Vol.35, n°139, 1996, and Proceedings of the 5th Int. Symposium on Fire Safety Science, pp.511-522, 1997.

NIST-114 (REV. 11-94) ADMAN 4.09	U.S. DEPARTMENT OF COMMERCE NATIONAL INSTITUTE OF STANDARDS AND TECHNOLOGY	(ERB USE ONLY)		
<h1 style="margin: 0;">MANUSCRIPT REVIEW AND APPROVAL</h1>		ERB CONTROL NUMBER	DIVISION	
		PUBLICATION REPORT NUMBER NIST-GCR-98-745		CATEGORY CODE
		PUBLICATION DATE April 1998	NUMBER PRINTED PAGES	
INSTRUCTIONS: ATTACH ORIGINAL OF THIS FORM TO ONE (1) COPY OF MANUSCRIPT AND SEND TO THE SECRETARY, APPROPRIATE EDITORIAL REVIEW BOARD.				
TITLE AND SUBTITLE (CITE IN FULL) Gas Generator Induced Flow and its Effect on Fire Flame Extinction				
CONTRACT OR GRANT NUMBER 70NANB7H0005		TYPE OF REPORT AND/OR PERIOD COVERED NIST-GCR-98-745		
AUTHOR(S) (LAST NAME, FIRST INITIAL, SECOND INITIAL) Joulain, Pierre & Torero, Jose		PERFORMING ORGANIZATION (CHECK (X) ONE BLOCK) <input type="checkbox"/> NIST/GAITHERSBURG <input type="checkbox"/> NIST/BOULDER <input type="checkbox"/> JILA/BOULDER		
LABORATORY AND DIVISION NAMES (FIRST NIST AUTHOR ONLY)				
SPONSORING ORGANIZATION NAME AND COMPLETE ADDRESS (STREET, CITY, STATE, ZIP) Universite de Poitiers & University of Maryland				
PROPOSED FOR NIST PUBLICATION <div style="display: flex; justify-content: space-between;"> <div style="width: 30%;"> <input type="checkbox"/> JOURNAL OF RESEARCH (NIST JRES) <input type="checkbox"/> J. PHYS. & CHEM. REF. DATA (JPCRD) <input type="checkbox"/> HANDBOOK (NIST HB) <input type="checkbox"/> SPECIAL PUBLICATION (NIST SP) <input type="checkbox"/> TECHNICAL NOTE (NIST TN) </div> <div style="width: 30%;"> <input type="checkbox"/> MONOGRAPH (NIST MN) <input type="checkbox"/> NATL. STD. REF. DATA SERIES (NIST NSRDS) <input type="checkbox"/> FEDERAL INF. PROCESS. STDS. (NIST FIPS) <input type="checkbox"/> LIST OF PUBLICATIONS (NIST LP) <input type="checkbox"/> NIST INTERAGENCY/INTERNAL REPORT (NISTIR) </div> <div style="width: 30%;"> <input type="checkbox"/> LETTER CIRCULAR <input type="checkbox"/> BUILDING SCIENCE SERIES <input type="checkbox"/> PRODUCT STANDARDS <input type="checkbox"/> OTHER _____ </div> </div>				
PROPOSED FOR NON-NIST PUBLICATION (CITE FULLY)		PUBLISHING MEDIUM		
<input type="checkbox"/> U.S. <input type="checkbox"/> FOREIGN		<input type="checkbox"/> PAPER <input type="checkbox"/> CD-ROM <input type="checkbox"/> DISKETTE (SPECIFY) _____ <input type="checkbox"/> OTHER (SPECIFY) _____		
SUPPLEMENTARY NOTES				
ABSTRACT (A 2000-CHARACTER OR LESS FACTUAL SUMMARY OF MOST SIGNIFICANT INFORMATION. IF DOCUMENT INCLUDES A SIGNIFICANT BIBLIOGRAPHY OR LITERATURE SURVEY, CITE IT HERE. SPELL OUT ACRONYMS ON FIRST REFERENCE.) (CONTINUE ON SEPARATE PAGE, IF NECESSARY.) Among the new technologies that are intended to replace Halon 1301 are Solid Propellant Gas Generators, (SPGGs). A facility that will serve to assess the performance of SPGGs has to be able to evaluate the combined effects of the gas discharge. Several alternatives have been proposed for an adequate flame and enclosure that will represent a "worst case" scenario for extinction and subsequent re-ignition. Among these alternatives is the recirculation zone induced by either a bluff body, a baffle, a backward facing step or a trench inside a wind tunnel. This report describes a preliminary evaluation of these configurations by means of a literature search and some preliminary computations using Large Eddy Simulation (LES) code developed at NIST. The literature review showed that the proposed configuration is adequate for the present application since it creates a recirculation zone with enhanced mixing, entrainment from the main stream and product evacuation. Preliminary calculations with the LES code showed qualitative agreement with reported experimental data while the literature showed that k-e codes seem inappropriate to model the recirculating zone. Different diagnostic techniques used for similar experiments are evaluated as candidates for characterization of the experimental facility.				
KEY WORDS (MAXIMUM OF 9; 28 CHARACTERS AND SPACES EACH; SEPARATE WITH SEMICOLONS; ALPHABETIC ORDER; CAPITALIZE ONLY PROPER NAMES) Fire suppression; gas generators; test methods				
AVAILABILITY <input checked="" type="checkbox"/> UNLIMITED <input type="checkbox"/> FOR OFFICIAL DISTRIBUTION - DO NOT RELEASE TO NTIS <input type="checkbox"/> ORDER FROM SUPERINTENDENT OF DOCUMENTS, U.S. GPO, WASHINGTON, DC 20402 <input checked="" type="checkbox"/> ORDER FROM NTIS, SPRINGFIELD, VA 22161		NOTE TO AUTHOR(S): IF YOU DO NOT WISH THIS MANUSCRIPT ANNOUNCED BEFORE PUBLICATION, PLEASE CHECK HERE. <input type="checkbox"/>		